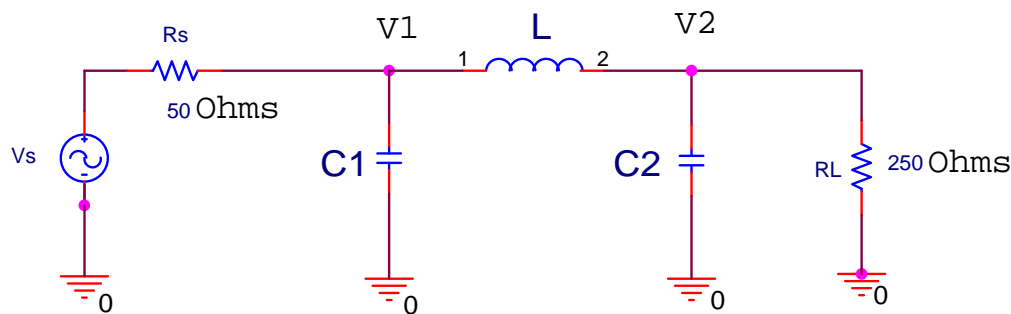


EE 411/511 Lab # 1

Π Match

Preliminary Work:

In this laboratory work you will try to design a matching network circuit known as “ Π Match”.



- 1) The matching should cover the frequency range 19.5-20.5 MHz. Match $R_p = 250$ Ohm to $R_{in} = 50$ Ohm at the center frequency of the match.
- 2) Find C_1 , C_2 and L values.
- 3) Design your coil. Use Wheeler's Formula. Note that for best Q select winding space = wire diameter.
- 4) Make SPICE simulation. Plot V_2 / V_s and V_1 / V_s from 1 MHz to 200 MHz (LOG scale).
- 5) This circuit is intended as the output matching circuit of a single transistor class-A or class-B amplifier driving a 50 ohm load. The small signal transistor gives the best performance when the 50 ohm load is reflected to the transistor as 250 ohms. The matching network should also suppress harmonics created by the amplifier. Make SPICE simulation again, this time 50 ohm side is the load side and 250 ohm side is driven by a current source for the frequency band covering the 2nd, 3rd, 4th and 5th harmonics of 20 MHz. Calculate how much each harmonic is attenuated (Take notice that we connect the source to 50 Ohm side for convenience).
- 6) Develop a LABVIEW program which measures and plots the frequency response of the matching circuit making use of the oscilloscope and the signal generator. You will be using this program to analyse your circuit at the lab.
- 7) Comment on how the source resistance of the signal generator will affect the measurement.

Lab Work:

- 1) Request your design values and an unetched PCB (Printed Circuit Board). You will solder these components on PCB.
- 2) Wind your coil. Measure its Q value.
- 3) Use soldering iron to solder the components on the copper side of PCB. Note that it will give best results when you use shorter leads.
- 4) Measure V_2/V_s and V_1/V_s ratios for a number of characteristic points on range $f = 1\text{MHz}$, to 30MHz . Plot these values over the simulation graph.
- 5) Now, use the network analyser to measure the input impedances of your circuit from both sides loading the other side properly and plot the impedances vs frequency for both cases.
- 6) Estimate R_{in} and Q of the circuit from the measurements.

Discussion:

Compare your experimental results with SPICE simulations. If there are differences, explain the reasons. Specifically note the effect of R_s on the Q of the circuit.